



Document details

< Back to results | 1 of 1

Export Download Print E-mail Save to PDF Add to List More... >

CFD Letters [Open Access](#)

Volume 11, Issue 5, 2019, Pages 37-49

Numerical investigation of flow field of a non-circular cylinder (Article)

Sajali, M.F.M.^a, Aabid, A.^a, Khan, S.A.^a ✉, Mehaboobali, F.A.G.^{b,c}, Sulaeman, E.^a

^aDepartment of Mechanical Engineering, Faculty of Engineering, International Islamic University Malaysia, Kuala Lumpur, 53100, Malaysia

^bDepartment of Mechanical Engineering, Bearys Institute of Technology, Mangalore, Karnataka, India

^cDepartment of Mechanical Engineering, Government Engineering College, Huvinahadagali, Karnataka, India

Abstract

The study of the base flow field or the flow field in the wake region is not accessible by using only a theoretical method; mostly this study has been done experimentally. The problem statement of this study is to know the effect of flow past of a non-circular cylinder on the drag. At high Reynolds number, the flow past a bluff body is characterized by a large wake zone. Therefore, drag reduction of the flow field is an interesting problem with a wide range of application. The present paper presents the numerical simulation of the flow field of a non-circular cylinder. The shielding effect of the square-plate front body on the flow field of drag reduction and the pressure distribution of a three-dimensional bluff body is simulated by using a numerical method. The results obtained from the simulation are compared with the experimental results. The results indicate that the side faces and the rear faces are subjected to low pressure, whereas the front face is experiencing high positive pressure. With this flow pattern, the pressure drag coefficient assumes a substantially significant value in the range of 1.0-1.42. Such a high value of drag coefficient is particularly valid for bluff bodies with noncircular cross-sections with sharp corners. © 2019 PENERBIT AKADEMIA BARU. All rights reserved.

SciVal Topic Prominence ⓘ

Topic: Nozzles | Mach number | Suddenly expanded

Prominence percentile: 63.870 ⓘ

Author keywords

ANSYS CFD D-shaped model Flowfield Pressure drag

Funding details

Funding sponsor	Funding number	Acronym
International Islamic University Malaysia		IIUM

Funding text

This work was supported by the Research Fund FRGS 17-036-0602 Ministry of Education through International Islamic University Malaysia.

Metrics ⓘ [View all metrics >](#)

14 Citations in Scopus

Field-Weighted
Citation Impact



PlumX Metrics

Usage, Captures, Mentions,
Social Media and Citations
beyond Scopus.

Cited by 14 documents

Modelling and analysis of convergent divergent nozzle with sudden expansion duct using finite element method

Aabid, A. , Chaudhary, Z.I. , Khan, S.A.
(2019) *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*

Estimation of damping derivatives for delta wings in hypersonic flow for straight leading edge

Monis, R.S. , Crasta, A. , Khan, S.A.
(2019) *International Journal of Mechanical and Production Engineering Research and Development*

Analysis of damping derivatives for delta wings in hypersonic flow for curved leading edges with full sine wave


Monis, R.S. , Crasta, A. , Faheem, M.
(2019) *International Journal of Engineering and Advanced Technology*

[View all 14 citing documents](#)

Inform me when this document is cited in Scopus:

[Set citation alert >](#)

[Set citation feed >](#)

 Khan, S.A.; Department of Mechanical Engineering, Faculty of Engineering, International Islamic University Malaysia, Kuala Lumpur, Malaysia; email:sakhan06@gmail.com
© Copyright 2019 Elsevier B.V., All rights reserved.

Find more related documents in Scopus based on:

Authors > Keywords >

< Back to results | 1 of 1

^ Top of page

About Scopus

What is Scopus
Content coverage
Scopus blog
Scopus API
Privacy matters



Language


日本語に切り替える
切换到简体中文
切换到繁體中文
Русский язык

Customer Service

Help
Contact us

ELSEVIER

[Terms and conditions](#)  [Privacy policy](#) 

Copyright © Elsevier B.V.  All rights reserved. Scopus® is a registered trademark of Elsevier B.V.

We use cookies to help provide and enhance our service and tailor content. By continuing, you agree to the use of cookies.

 RELX